

NETWORK FLOW SIMULATION OF FLUID TRANSIENTS IN ROCKET PROPULSION SYSTEMS

Alak Bandyopadhyay
Computer Science
Alabama A & M University
Normal, AL

Brian Hamill^{*}, Narayanan Ramachandran^{**} and Alok Majumdar^{*}
NASA/Marshall Space Flight Center
^{**}Jacobs Technology Inc
Huntsville, AL

ABSTRACT

Fluid transients, also known as water hammer, can have a significant impact on the design and operation of both spacecraft and launch vehicle propulsion systems. These transients often occur at system activation and shutdown. The pressure rise due to sudden opening and closing of valves of propulsion feed lines can cause serious damage during activation and shutdown of propulsion systems. During activation (valve opening) and shutdown (valve closing), pressure surges must be predicted accurately to ensure structural integrity of the propulsion system fluid network.

In the current work, a network flow simulation software (Generalized Fluid System Simulation Program) based on Finite Volume Method has been used to predict the pressure surges in the feed line due to both valve closing and valve opening using two separate geometrical configurations. The valve opening pressure surge results are compared with experimental data available in the literature and the numerical results compared very well within reasonable accuracy ($< 5\%$) for a wide range of inlet-to-initial pressure ratios. A Fast Fourier Transform is performed on the pressure oscillations to predict the various modal frequencies of the pressure wave.

The shutdown problem, i.e. valve closing problem, the simulation results are compared with the results of Method of Characteristics. Most rocket engines experience a longitudinal acceleration, known as “pogo” during the later stage of engine burn. In the shutdown example problem, an accumulator has been used in the feed system to demonstrate the “pogo” mitigation effects in the feed system of propellant. The simulation results using GFSSP compared very well with the results of Method of Characteristics.

INTRODUCTION

Fluid transients (also known as Water hammer) have significant impact in design and operation of spacecraft and launch vehicle propulsion systems. The pressure rise due to sudden opening and closing of valves of propulsion feed line can cause serious damage during activation and shutdown of propulsion systems. Pressure surge occurs when either a propellant feed line system is opened or closed suddenly by using control valves. The accurate prediction of pressure surge is quite important from structural integrity point of view of the propulsion systems. Most rocket engines experience a longitudinal instability, known as “Pogo” during the later stage of engine burn. To mitigate pogo oscillation, an accumulator is used in the feed system of liquid propellant to detune the vibration coupling of the feed line with the vehicle structural frequencies.

There have been numerous studies to predict the pressure surge in pipeline-reservoir systems. Hearn [1] has studied the effect of valve opening in propellant loading line from a reservoir both analytically and experimentally. The Method Of Characteristic (MOC) is one of the most widely used semi-analytical method for water hammer surge prediction [2, 3]. This method is based on solving ordinary differential equation along line of characteristics. MOC, however, is not particularly suited for typical fluid network with branching, solid-to-fluid heat transfer and phase change. Majumdar and Flachbart [4] have used the network flow analysis simulation based on finite volume method to compute the fluid transients of long cryogenic pipeline and they have compared their results with that of the Method of Characteristic simulation results.

In most recent study, Lee [5] has studied the fluid transient problem in predicting the pressure rise in entrapped air in a horizontal pipe experimentally. This problem is an example of sudden opening of valve and sudden acceleration of liquid mass into the air space. The presence of air in the pipe causes excessive pressure rise in the pipe.

In the present study, two separate problems representing pressure rise due to valve opening and valve closing are solved numerically using the finite volume based network fluid analysis simulation program (Generalized Fluid System Simulation Program) and the results are compared with the experimental results of Lee [5] and semi-analytical MOC results respectively. A Fast Fourier Transform analysis has been conducted to convert the pressure-time characteristics into a frequency domain to predict the frequency band of the pressure oscillations.

PROBLEM DESCRIPTION

Two separate problems are considered for the study. The first problem is an example of a valve opening problem where liquid is allowed to enter a pipe line having entrapped air by opening a ball valve. The second problem is an example of sudden valve closing. These two problems are described in brief below.

a) Valve Opening Problem

A long pipe is attached to a reservoir at one end containing liquid water and closed at the other end as shown in the figure 1. The liquid water and entrapped air regions in the pipe are separated by a ball valve. The dimension of the pipe and other controlling parameters such as reservoir-to-air pressure ratio, length of air column etc are taken identical to that used by Lee [5] so that the numerical results can be compared to the experimental data of Lee [5]. The ball valve is opened from a 0% opening to 100% opening by controlling the angle of the ball valve and this is shown in figure 2. The reservoir pressure is a few times higher than the pressure of the entrapped air (air is assumed to be at atmospheric pressure). The ratio of reservoir pressure to the initial pressure (P_R) varies in the range of 2 to 7, i.e. the reservoir pressure (p_R) range being 29.4 psi to 102.9 psi. The dimension of the pipe and other parameters are identical to that used by Lee [5] in his experimental work, so that the current study results can be compared with Lee's experimental results. Apart from the initial pressure ratio, another controlling parameter is the ratio of initial length of the entrapped air column to the total length of the pipe ($\alpha = L_g/L_T$). The

initial length for the water volume in the pipe (L_l) is fixed to 20 ft, and initial length of air column in the pipe (L_g) varies from a low of 1.23 ft to 16.23 ft, the value of α ranging from 0.0579 to 0.448 respectively. The pipe diameter is 1.025 inches. The entrapped air and water is initially at 14.7 psia and 60 F.

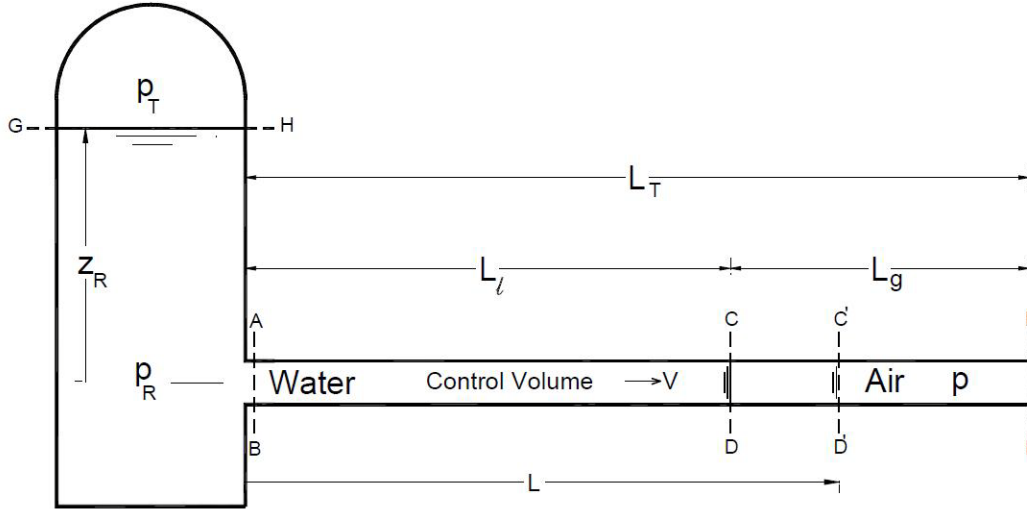


Figure 1. Schematic of the Water Pipe with entrapped Air [5]

The ball valve does not open until about 0.15 sec, and gradually starts opening and it opens 100% at about 0.4 second. Figure 2 shows the ball valve angle position with time and 0 degree refers to full closed and 90 degree refers to full open position. In the present numerical model this is accounted for by providing valve area change history.

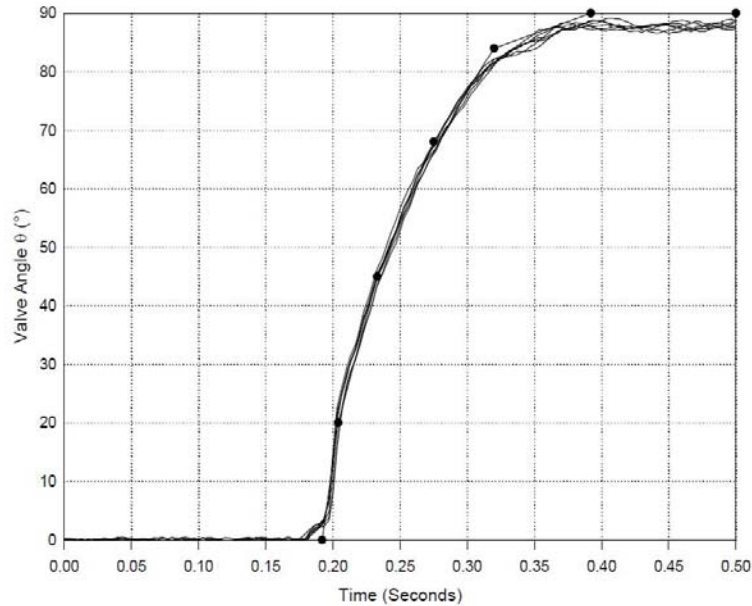


Figure 2. Ball Valve Angle Change with Time [5]

b) Sudden Valve Closure

A long pipe line carrying liquid oxygen is subjected to sudden valve closure. An accumulator containing helium gas is placed upstream of the valve to reduce the amplitude and frequency of the pressure oscillation. A schematic diagram of the system is as shown in figure 3. The pipe is 400 ft long and 1/4 inch diameter. The accumulator is placed just before the valve.

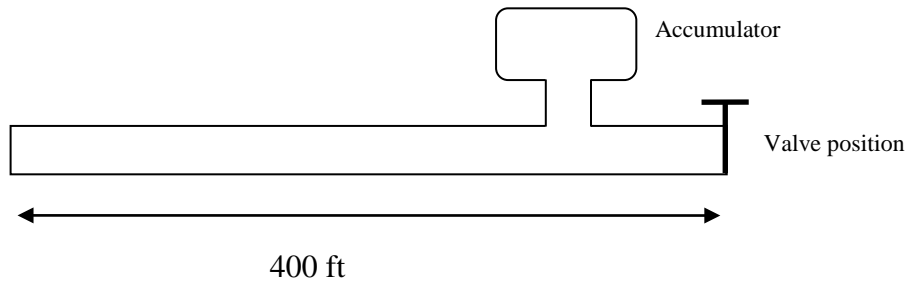


Figure 3. Schematic diagram of the valve closing problem with accumulator.

In the present study, the effect of the accumulator is studied by removing it from the feed line and placing it back just before the valve. It is assumed that the accumulator is charged with gaseous helium, and represents a pogo suppressing device in the feed line.

RESULTS AND DISCUSSION

In this section the numerical method based on finite volume principle will be described in brief followed by the results for two separate problems. The generalized fluid simulation software program (GFSSP) is based on the numerical method described and instrumental in solving both the problems of valve opening and closing. The results of the valve opening problem are compared with the experimental data of Lee [5] and Lee and Martin [6] and the results of the valve closing problem are compared with the Method of Characteristics.

NUMERICAL MODELING

GFSSP is a finite volume based network analysis software [7], which resolves a given flow network into a finite number of nodes interconnected by branches. The nodes consist of a) Boundary Nodes where pressure, temperature and species concentrations are specified, and b) Internal Nodes where pressure, temperature and species concentrations are calculated by solving mass, energy and species conservation equations respectively. The momentum equations are solved in the branches to calculate the flowrate entering or leaving the node. The flow could be steady or unsteady, compressible or incompressible, and with or without heat transfer. Thermodynamic properties at each node are computed by GASP [8] or WASP [9]. The system of equations is solved by a combination of Newton-Raphson and successive substitution method. GFSSP has a user friendly Graphical User Interface, VTASC that allows user to develop GFSSP model of complex network by point and click paradigm.

GFSSP MODELS:

a) Numerical Model for the Valve Open Problem:

The physical problem shown in Figure 1 is numerically modeled by dividing the region from the exit of the reservoir to the valve by using a total of 12 nodes, out of which node 1 is the boundary node as shown in Figure 3. The restriction is used to control the flow of water into the pipe. The volume of node 12 is adjusted as the volume of water expands and the corresponding surrounding air volume reduces. This is done to adjust mutual change in volume of the water column and the entrapped air.

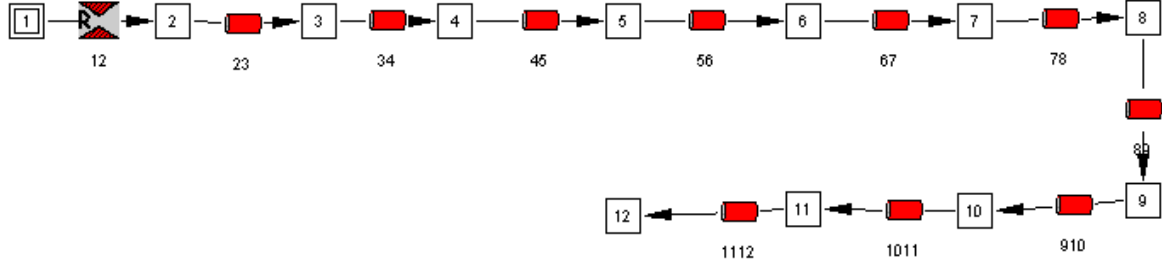


Figure 4. Finite Volume Model of the Flow Network

This is done in GFSSP through the user subroutine by adding a volume source for the working fluid (water in this case). This volume source is computed based on the relation given below.

Change in node volume:

As soon as the valve is opened, the water in the pipe is going to compress the entrapped air, and this volume increase of water is adjusted as a volume source for node 12. The air volume would reduce by the same amount. Using ideal gas assumption and thermodynamic relation, the volume source for the working medium (water) is given as:

$$\Delta V_{\text{water}} = -\Delta V_{\text{air}} = m_{\text{air}} R_{\text{air}} T_{12} \Delta p / p^2$$

Where, Δp = change in pressure at node 12 with time = $p - p^*$, where both p and p^* represent pressure computed at node 12, at the current and previous time steps respectively.

Computation of Air pressure:

The entrapped air is considered stationary and so the momentum and other conservation equations are not solved for air. The bulk pressure in the air domain is computed by using thermodynamic relations. As the whole process is very fast (valve opening is about 4 seconds), the compression in the air is assumed to be adiabatic.

$$p_{\text{air}} = \frac{p_{\text{air}}^* V_{\text{air}} + p_{\text{air}}^* V_{\text{air}}^* \left[\left(\frac{V_{\text{air}}^*}{V_{\text{air}}} \right)^{\gamma-1} - 1 \right]}{V_{\text{air}} + \Delta V_{\text{air}}}$$

Additional adjustment for momentum source at node 12:

The pressure developed within the air will try to prevent water column movement and this resistance needs to be incorporated in the model and this is done as a momentum sink to the last control volume (branch 11-12) of the computation domain.

Momentum Source for branch 1112 = $-(p_{\text{air}} - p_{12})A$

where, A is the cross-sectional area.

b) Numerical Model for the Valve Closure Problem

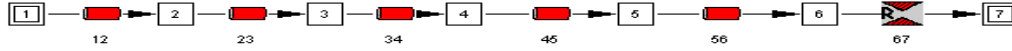


Figure 5. GFSSP Model of Valve Closure Problem

Figure 5 shows a flow network consisting of a 400 feet pipeline modeled with five internal nodes (2, 3, 4, 5 and 6) and five (12, 23, 34, 45 and 56) branches. Branch 67 represents a valve which closes in 0.1 second. Node 1 and 7 represents boundary nodes representing supply and outlet pressures respectively. Pressures and temperatures are computed in the internal nodes by solving the mass and energy conservation equations and flowrate are computed in branches by solving the momentum conservation equations. In this simulation the accumulator was modeled in node 5 as a node with variable volume. Part of node volume is occupied by Helium gas; the gas volume changes due to compressibility. As the pressure wave travels along the pipeline, the gas volume fluctuates to provide compliance to the system.

RESULTS FOR VALVE OPEN CASE

The computed results of GFSSP is compared with that of Lee and Martin [6] for three different geometrical configurations: (a) $\alpha = 0.448$ (b) $\alpha = 0.195$ and (c) $\alpha = 0.058$. The table below shows the values of air column length as a function of alpha.

α	L_g	L_T
0.4480	16.23	36.23
0.1952	4.85	24.85
0.0579	1.23	21.23

Table 1: List of Geometrical Parameters for Numerical Case Study

GRID INDEPENDENCE TEST: A grid independence study was done for the numerical solution to see the effect of grid size. For this purpose, a case study was done with the inlet to initial pressure ratio $P_R = 7$ and air length to total length ratio $\alpha = 0.448$. Figure 6(a) shows that the entrapped air pressure does not change appreciably when the number of nodes is doubled.

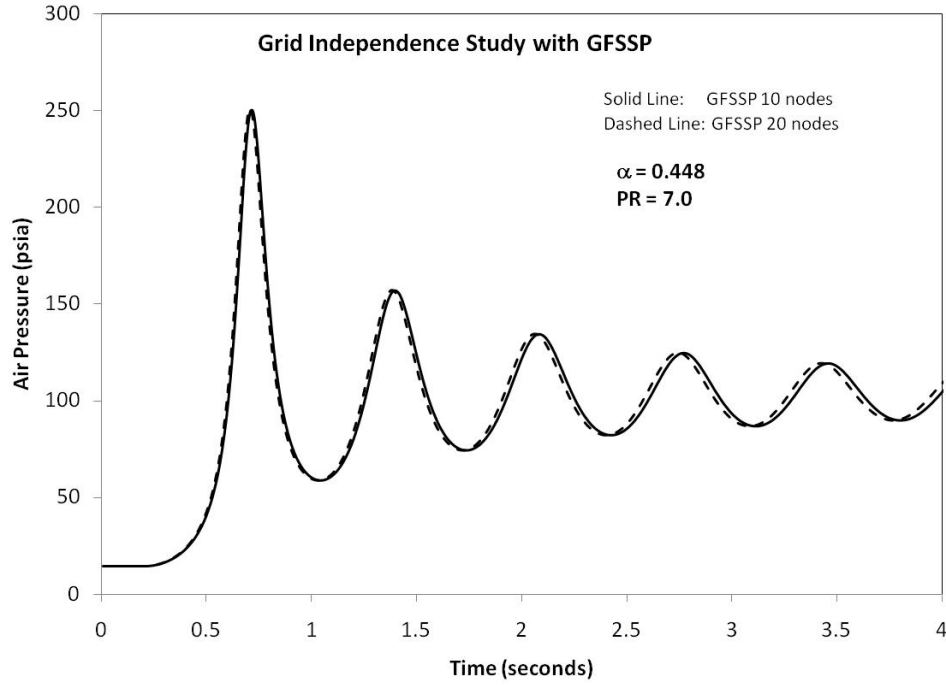


Figure 6(a). Grid Independence study of numerical solution on air pressure

TIMESTEP INDEPENDENCE:

In order to get a time step independent solution of the problem, the simulation was carried out with time steps of 0.01, 0.005 and 0.0025 sec. These time steps satisfy the Courant condition of $\text{Courant No.} = U\Delta\tau/\Delta x$ where U is characteristic velocity, $\Delta\tau$ is the time step and Δx is the spatial interval. CFL condition needs that the Courant no. should be less or equal to 1 for a stable solution. Figure 6(b) shows that a reduction of time step from a time step of 0.005 s does not change the numerical results considerably and hence for the entire simulation, a time step of 0.005 sec is adequate for reasonably accurate solution.

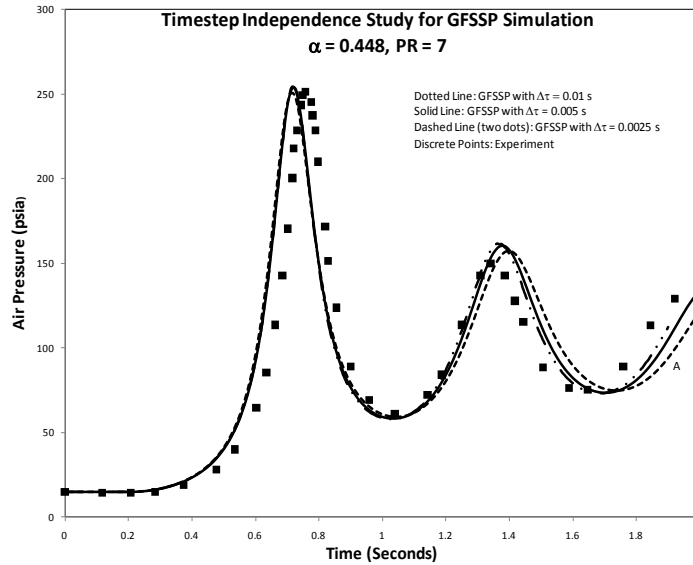
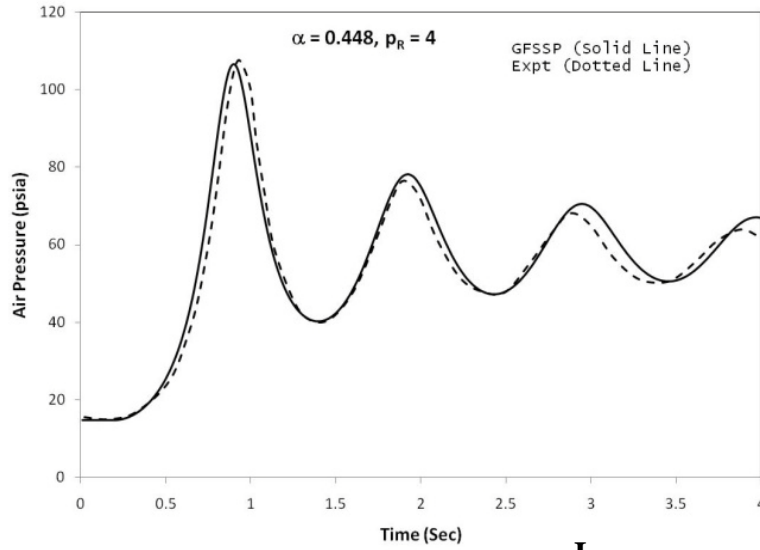


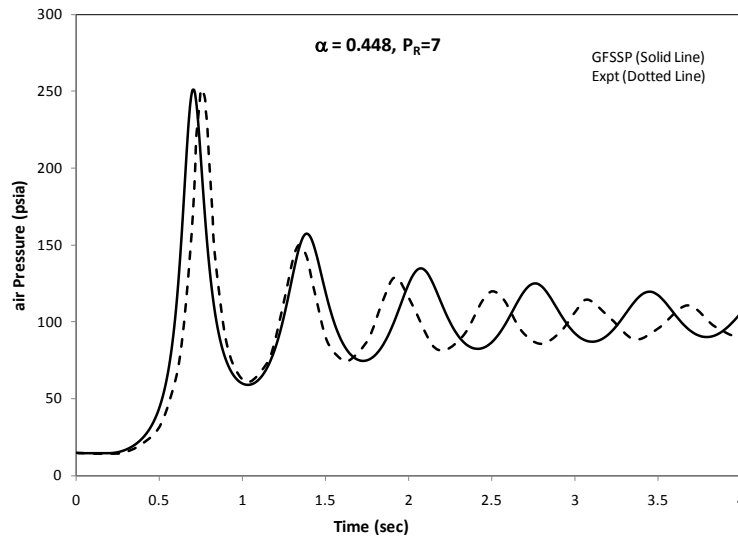
Figure 6(b). Time independence study of the numerical solution.

VALIDATION WITH EXPERIMENTS:

The numerical results of GFSSP are compared with the experimental data of Lee [5] for comparing the pressure developed in the entrapped air for three different geometrical configurations (a) $\alpha = 0.445$ with Pressure ratio (P_R) of 4 and 7, (b) $\alpha = 0.1952$ with Pressure ratio (P_R) of 4 and 7 and (c) $\alpha = 0.058$ with Pressure ratio (P_R) of 3 and 5. For same pressure ratio, the maximum pressure rise will be for smaller α . Figure 7 show the plot of air pressure variation with time for case (a) $\alpha = 0.448$ with two different pressure ratios (I) $P_R = 4$ and (II) $P_R = 7$.



I



II

Figure 7. Validation of air pressure for (I) $P_R = 4$ and (II) $P_R = 7$ at about 45% initial air volume ($\alpha \approx 0.45$)

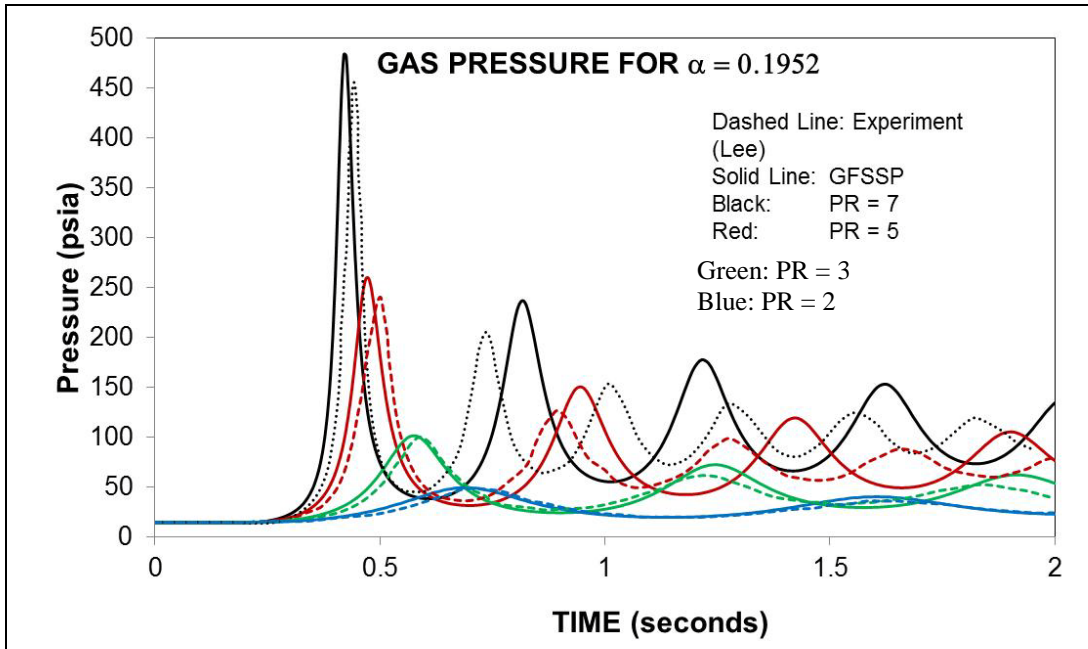


Figure 8. Air Pressure at various Pressure ratios for case (b), $\alpha \approx 0.2$ (20% air)

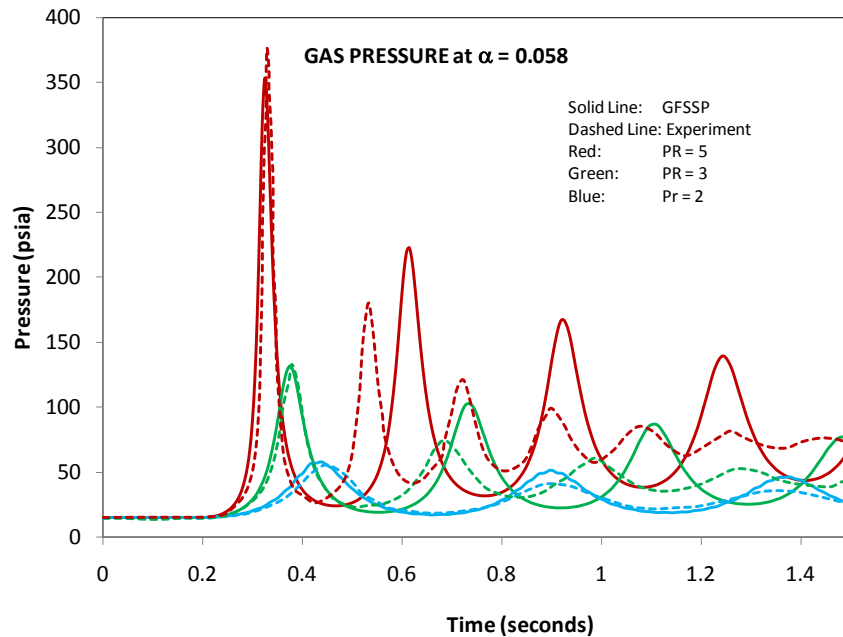


Figure 9. Air Pressure at various Pressure ratios for case (c), $\alpha \approx 0.06$ (6% air)

Figures 8 and 9 show the pressure rise in the entrapped air when relative less air is present in the entrapped pipe initially (approximately 20% and 6% by volume respectively). The air pressure rises to approximately 260 psia and 380 psia respectively when the reservoir pressure is 73.5 psia for 20% and 6% initial air respectively. The comparison with the experimental data (represented by dotted lines in the figures) show quite good agreement as far as amplitude of the pressure oscillations are concerned even at higher reservoir pressure, and

the agreement is excellent (within 5% or so) at medium and low reservoir pressure (44 psia or less). The discrepancy at higher pressure could be due to the rigid pipe assumption and due to not accounting for pipe deformation.

WATER PRESSURE AND FAST FOURIER TRANSFORM

The pressures in the water column at different nodes (spatial locations) are plotted as a function of time as shown in figure 10 below. Node 2 is the close to the entry of pipe from the reservoir and node 11 is almost at the end of the pipe line. The pressure oscillation is due to the water hammer effect, and also due to compressed air pushing the water column back from expanding. The frequency of pressure oscillations can be better analyzed by converting the transient pressure response to a frequency domain as shown in Figure 11.

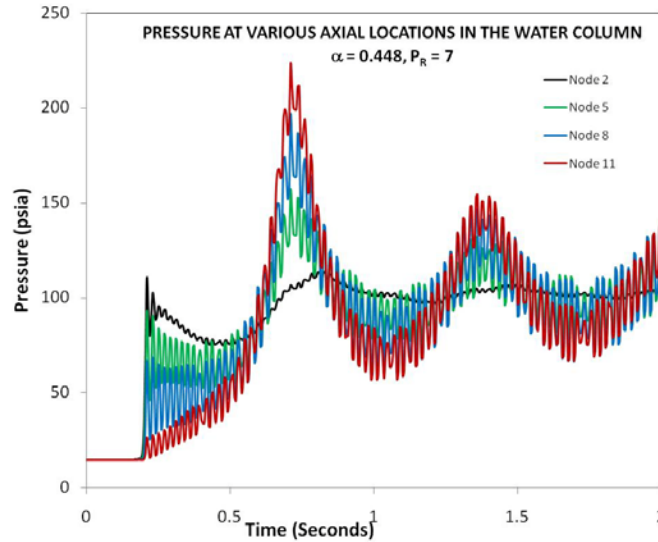


Figure 10. Pressure Transients at various spatial locations

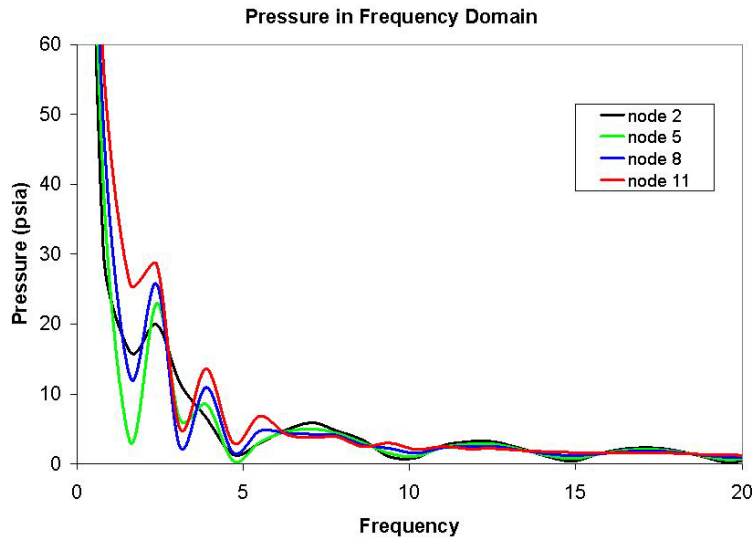


Figure 11. Nodal Pressure response in Frequency Domain.

Figure 11 shows frequency band of the primary, secondary and other higher order modes. This transformation is being done by using a First Fourier Transformation of the pressure-time response using the following transformation function.

$$X(e^{i\Omega}) = \sum_{n=-\infty}^{\infty} x[n]e^{-i\Omega n}$$

Here, $x[n]$ represents the time domain function and $X[\Omega]$ in the frequency domain. The real part of the complex field represents the amplitude and the imaginary part gives the frequency. The lower frequency represents the primary mode oscillation frequency and the higher ones for the other modes.

RESULTS FOR VALVE CLOSURE PROBLEM

Figure 12 shows the GFSSP simulation results for the valve closing problem (no accumulator) with two different grids (20 and 40) and results are compared with the Method of Characteristic results. Further increase in number of grid points to 80 nodes did not change the numerical results appreciably. The amplitude and frequency matched quite well between the network flow analysis (GFSSP) simulation with the semi-analytical Method of Characteristic results.

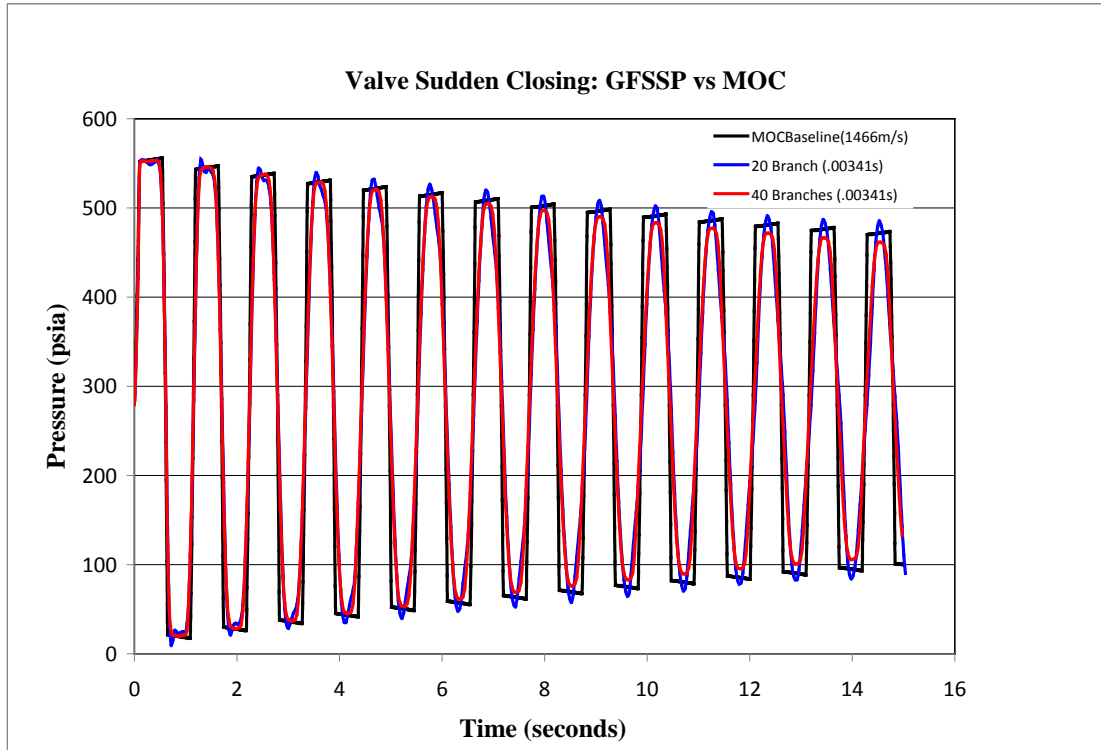


Figure 12: Comparison of numerical results of GFSSP simulation with MOC.

Figure 13 shows the effect of having accumulator in the pipe line, and it has been shown that the accumulator dampens the pressure pick and the pressure oscillation eventually completely zeroed down.

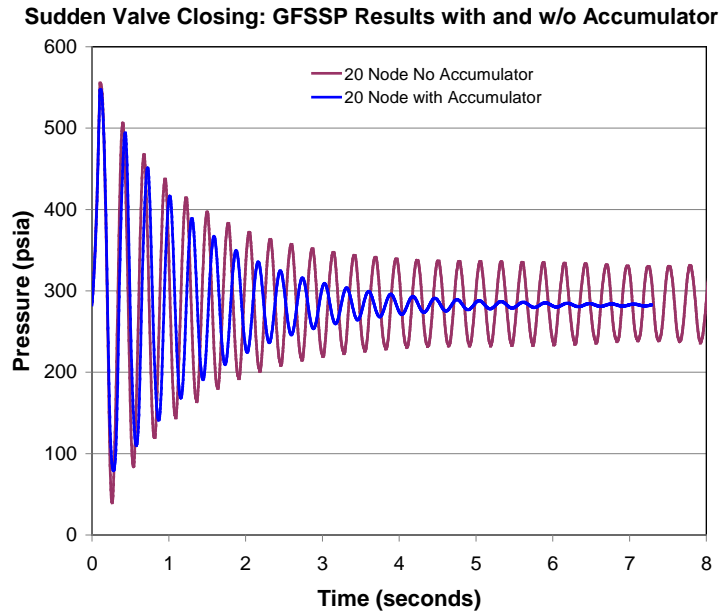


Figure 13: Effect of Accumulator on Pressure Rise

SUMMARY AND CONCLUSIONS

This paper demonstrates that the finite volume based network flow analysis method implemented in GFSSP can accurately predict fluid transient during rapid opening and closing of valve. It can also model the compliance caused due to the compressibility of the gas. The simulation results from GFSSP compared reasonably well with the experimental data for the valve opening problem and with that of method of characteristics for the valve closing problem. For the valve opening problem with entrapped air in the pipe, the agreement of computed results with the experimental data of Lee [5] is excellent at high ratio of air to water, and in particular when the ratio of inlet pressure to initial air pressure is low (≤ 4). The deviation between experiment and prediction at higher pressure ratio is due to the assumption of rigid pipe while the experiments were carried out in plastic tubes that may have added to the compliance. Future work will include fluid-structure interaction as well as the effect of heat transfer and phase change in fluid transients.

REFERENCES

1. Hearn, H.C., "*Development and Application of a Priming Surge Analysis Methodology*", Proc. 41st AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, July 2005, Tucson, AZ, pp. 1-10.
2. Wylie, E.B., and Streeter, V., *Fluid Transients*, FEB Press, Ann Arbor, 1982, pp 13-61.
3. Moody, F.J., *Introduction to Unsteady Thermo fluid Mechanics*, John Wiley & Sons, 1990, pp 408-422.
4. Majumdar, A.K., and Flachbart, R.H., "*Numerical Modeling of Fluid Transients by a Finite Volume Procedure for Rocket Propulsion Systems*", Proc. ASME FEDSM'03, 4th ASME/JSME Joint Fluids Engg. Conference, Hawaii, 2003, pp 1-8.
5. Lee, N.H., "Effect of Pressurization and Expulsion of Entrapped Air in Pipelines", Ph.D. thesis, Georgia Tech., August 2005.
6. Lee, N.H., and Martin C.S., "*Experimental and analytical investigation of entrapped air in a horizontal pipe*", Proceedings of the 3rd ASME/JSME Joint Fluids Engineering Conference San Francisco, CA, 18–23 July (1999).
7. Majumdar, A.K., "*Generalized Fluid System Simulation Program, Version 5.0 – Educational*", NASA/TM – 2011-216470, August 2011.
8. Hendricks, R. C., Baron, A. K., and Peller, I. C., "GASP - A Computer Code for Calculating the Thermodynamic and Transport Properties for Ten Fluids: Parahydrogen, Helium, Neon, Methane, Nitrogen, Carbon Monoxide, Oxygen, Fluorine, Argon, and Carbon Dioxide", NASA TN D-7808, February, 1975.
9. Hendricks, R. C., Peller, I. C., and Baron, A. K., "WASP - A Flexible Fortran IV Computer Code for Calculating Water and Steam Properties", NASA TN D-7391, November, 1973.